RECLAMATION

Managing Water in the West

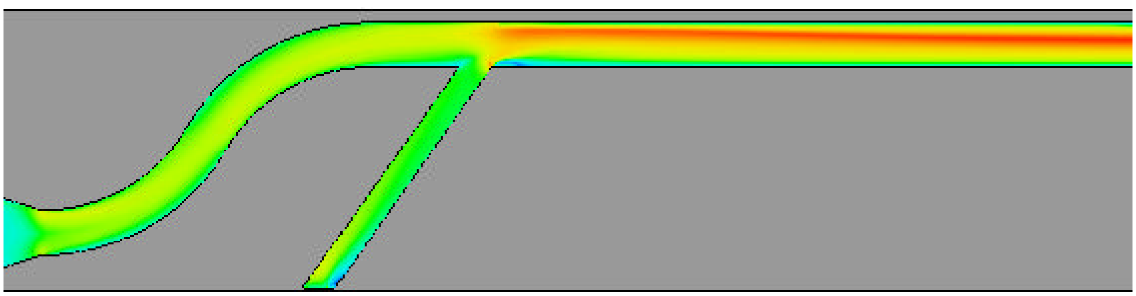
Technical Memorandum

All-American Canal and Drop 2 Outlet Canal Confluence

Computational Fluid Dynamics Analysis of Operating Conditions All-American Canal System

Boulder Canyon Project

Lower Colorado Region



U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Water Resources Research Laboratory

Denver, Colorado **August 2007**

Mission Statements

The mission of the Department of the Interior is to protect and provide access to our Nation's natural and cultural heritage and honor our trust responsibilities to Indian Tribes and our commitments to island communities.

The mission of the Bureau of Reclamation is to manage, develop, and protect water and related resources in an environmentally and economically sound manner in the interest of the American public.

Acknowledgments

Special thanks to Morris "Rudy" Campbell for generating the Stereolithography files for this study and Warren Frizell for Peer review.

Disclaimer

Executive Summary

Project Description

Water from the Drop 2 storage reservoir will be returned to the All American Canal though an outlet system that features an inverted siphon, transitioning into a short (1,100 ft) canal section that confluences with the All American Canal downstream of the exiting All American Canal Drop #2 (Drop #2). Hydraulics in this return transition is the focus of the studies reported in this document. The design flow of the outlet canal is 1,800 ft^3/s. [i]

Study Introduction

Computational Fluid Dynamics (CFD) software was used to investigate the confluence of this new canal section and the All American Canal. Headloss, waves, and left bank water surface elevation (canal bank run up) were investigated for several operational variations (Table 1) of the new structure. In addition, sedimentation potential and safety concerns for swimmers in the canal were also evaluated. All simulations assumed a steady and uniform velocity distribution at either/or the Drop 2 and siphon outfalls. Any influences from gate operations or power generation were assumed to have no direct influence on the confluence and were not considered.

Results

- The CFD simulations showed changes to water surface elevation (WSEL) due to the confluence when not discharging flows into the All American Canal to be insignificant as determined by the slope of the water surface elevation through the zone.
- The maximum water surface elevation change that can be attributed to confluence operation is minimal for all conditions tested. Comparisons of water surface elevations are displayed in Table 1.

Table 1. Water surface elevations at Drop 2. The four simulated flow conditions and purpose are shown. Each simulation use the water surface elevation 121.4 feet at exit of model (X=5,469). Simulations did not include the effects of outflow from Drop 2.

Discharge (ft^3/s)	Location	Condition or Purpose of Simulation	WSEL at Drop 2 (feet)
7,400	All American Canal	Maximum inflow and outflow of the All American Canal section	122.4
5,600	All American Canal	Maximum Outlet Canal flow with maximum downstream All American Canal flow	122.5
1,800	Outlet Canal		
5,600	All American Canal	Comparison to other conditions	122.1
1,800	Outlet Canal	Maximum Outlet Canal flow	121.5

- Waves generated by the joining flows resulted in maximum wave heights of about 0.9 inches in the vicinity of the confluence. (The wave simulations did not include wind driven waves.)
- Left bank water surface elevation (bank run up) attributed to the confluence operating was only 0.05 feet.
- Sediment deposition is likely to occur in the Outlet Canal of the Drop 2 reservoir when it is not operating and there is flow in the All American Canal. The amount of sediment that could be deposited would likely be very dependent on the suspended sediment load carried in the main canal flow. When water is transferred from the Drop 2 reservoir to the All American Canal at near maximum discharge, any deposited sediment would likely be resuspended and cleaned out.
- There are minor safety-related concerns regarding swimmers in the canal caused by secondary currents within the canal. The transition does not appear to have much influence of these conditions but rather the S-shaped

bend upstream from the confluence dominates the flow patterns that create slight down-welling. Standard safety ladders would be appropriate.

Computational Fluid Dynamics Modeling

There are many steps required to develop an appropriate Computational Fluid Dynamics (CFD) model. These include development, refinement, testing of the grid, boundary conditions, model extents, and obstacles (structures) for the CFD program.

CFD Program Description

The CFD program FLOW-3D Version 9.2 by Flow Science Inc., was used for these studies. FLOW-3D [ii] is a finite difference/volume, free surface, transient flow modeling system, developed to solve the Navier [iii]-Stokes equations [iv] in three spatial dimensions.

The finite difference equations are based on an Eulerian mesh of non-uniform hexahedral (brick shaped) control volumes using the Fractional Area/Volume (FAVOR) [v] method. Free surfaces and material interfaces are defined by a fractional volume-of-fluid (VOF) function [vi]. FLOW-3D uses an orthogonal coordinate system as opposed to a body-fitted system.

Flow-3D can have a single nested mesh block, one completely contained by another, adjacent linked mesh blocks, or a combination of nested and linked mesh blocks. The final simulations of this study used a nested mesh block to refine waves simulated along the left bank.

Simulation Parameters

Final simulations used

- Free surface,
- Concrete roughness of 0.02 feet,
- Renormalized group turbulence model,

- Dynamic viscosity of 2.25e-5 lb•s/ft^2,
- Fluid density of 1.937 slugs/ft^3,
- Water surface elevation 121.414 feet at exit of model (X=5,469).
- Line Successive Over-relaxation implicit solver in the X- and Z-directions, and
- First order advection method.

The physical domain of the model (Figure 1) extended from the outfall of Drop 2 to 5,469 feet downstream (measured parallel to the All American Canal). The model also included the canal confluence from the Drop 2 storage reservoirs. Actual elevations were used in the model while X=0 and Y=0 was set at the Drop 2 outfall.

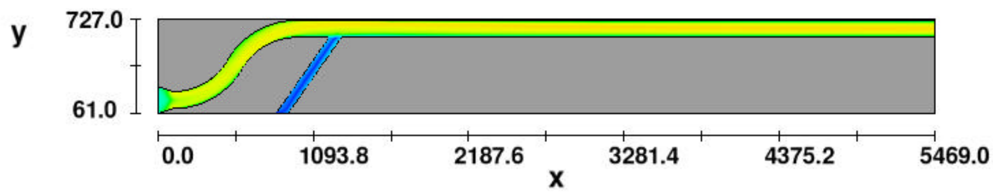


Figure 1. The physical domain of the model. Full extents of the model are shown here in the XY plane. Z values ranged from elevation 101 feet to 128 feet and used actual elevation values.

In the final simulations, the outer mesh-block cells were 3 feet on each side and contained 4,101,750 cells. The nested mesh-block cells were 1 foot on each side and contained 488,160 cells and set along the left bank as shown in Figure 2

The design of the outfall of Drop 2 was not known at the time of the modeling so an even inflow distribution was used. It was expected that turbulence directly caused by Drop 2 would be dampened out by the end of the S-shape bend. Accordingly, simulated wave heights in the bends are likely too small and are not reported.

In all 4 simulations, flow into the simulations at the Drop 2 and/or siphon outfalls used source objects which had a steady and even flow distribution. The top of the source objects were placed below the water surface and were capped with another object to prevent vertical flow at the inflow source and causing all of the flow to be directed downstream. Source object allow precise discharge control as opposed to a specified velocity boundary which can cause the discharge to be mesh dependent.

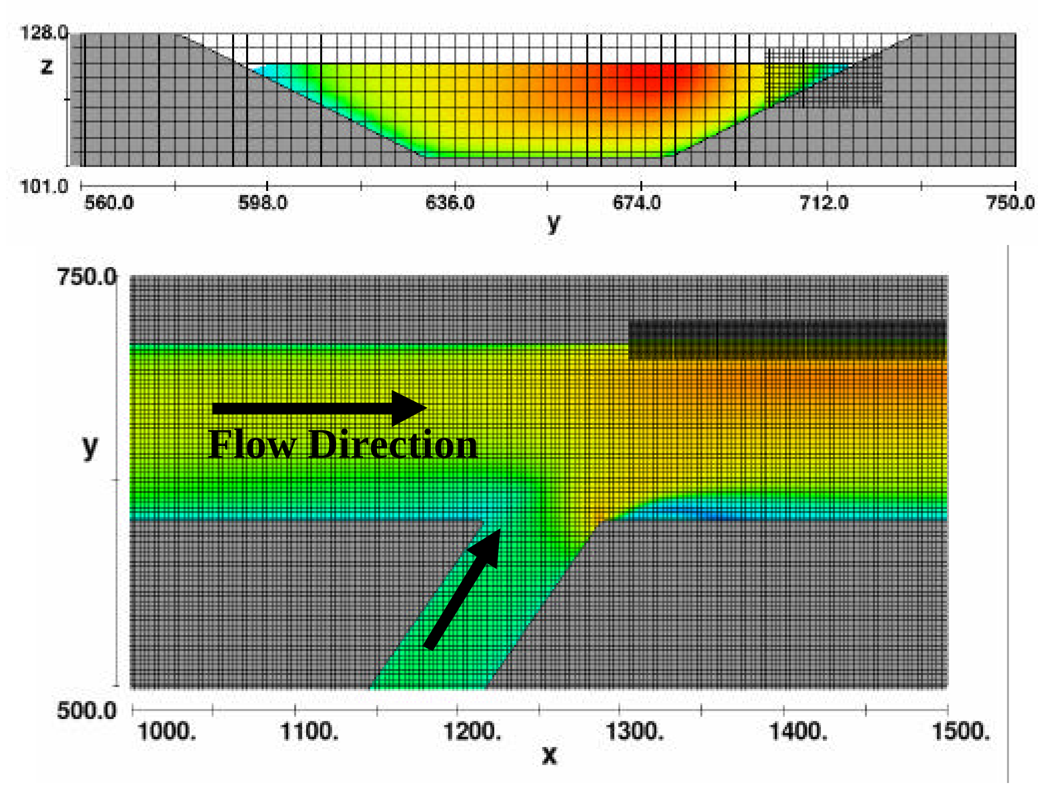


Figure 2. Mesh lines in a portion of the YZ and XY planes. The top image shows a profile looking upstream with the nested mesh block on the left bank. The lower image is a horizontal slice at elevation 117.5 feet and displays the upstream portion of the nested mesh block. The outer mesh block used cells with 3 foot sides, while the nested mesh block used cells with 1 foot sides. Actual elevations were used for Z values. Waves with a surface area greater than 1 foot by 1 foot were captured.

In all 4 simulations, the downstream boundary was set to a pressure boundary with a fluid height set to 121.41 feet. This water surface elevation was calculated by applying the normal depth of 14.72 feet to the elevation of the canal invert at the model's exit at 106.69 feet.

The downstream boundary was placed farther than would be expected from the confluence to reduce boundary-feed back in the confluence. Using a downstream boundary close to the confluence caused additional surface waves and required longer simulations time to settle out the waves. Boundary-feed back is typical while simulating this type of boundary and sub-critical flow. A wave's valley below elevation 121.41 feet traveling downstream is reflected by a wave's peak higher than elevation 121.41 feet traveling upstream. To insure the waves did not affect the zone of concern, the model was lengthened to 5,469 feet. The surface waves would have a very minor effect on headloss through the model.

Simulations of the Confluence Transition

The design capacity of the All American Canal was 7,400 ft^3/s while the design capacity of the Outlet Canal was 1,800 ft^3/s. Four flow conditions were simulated to identify headloss, wave heights, and adverse flow conditions. They were also used to predict where sediment deposition may occur. The four simulated conditions are listed in Table 2. The frequency of these flow conditions were not known at the time of the study.

Table 2. Simulated Cases. The four simulated flow conditions are shown.

Case	Discharge (ft^3/s)	Location
1	7,400	All American Canal
2	5,600	All American Canal
	1,800	Outlet Canal
3	5,600	All American Canal
4	1,800	Outlet Canal

Case 1 - 7,400 ft^3/s All American Canal

Case 1 had 7,400 ft^3/s entering the All American Canal and no flow entering from the Outlet Canal. Figure 3 shows the plan view of the conditions. Figure 4 shows sectional views of the flow and reveals that secondary currents are created by the bends that extend past the confluence.

Water surface elevations were recorded at 0.5 second intervals along the left bank at X=1400 feet and X=1600 feet (where the converging flows are expected to have the highest turbulence) and results are displayed in Figure 5. The CFD time step varied around 0.06 seconds. It is noted that the simulation did not include wind or wind effect of wave action.

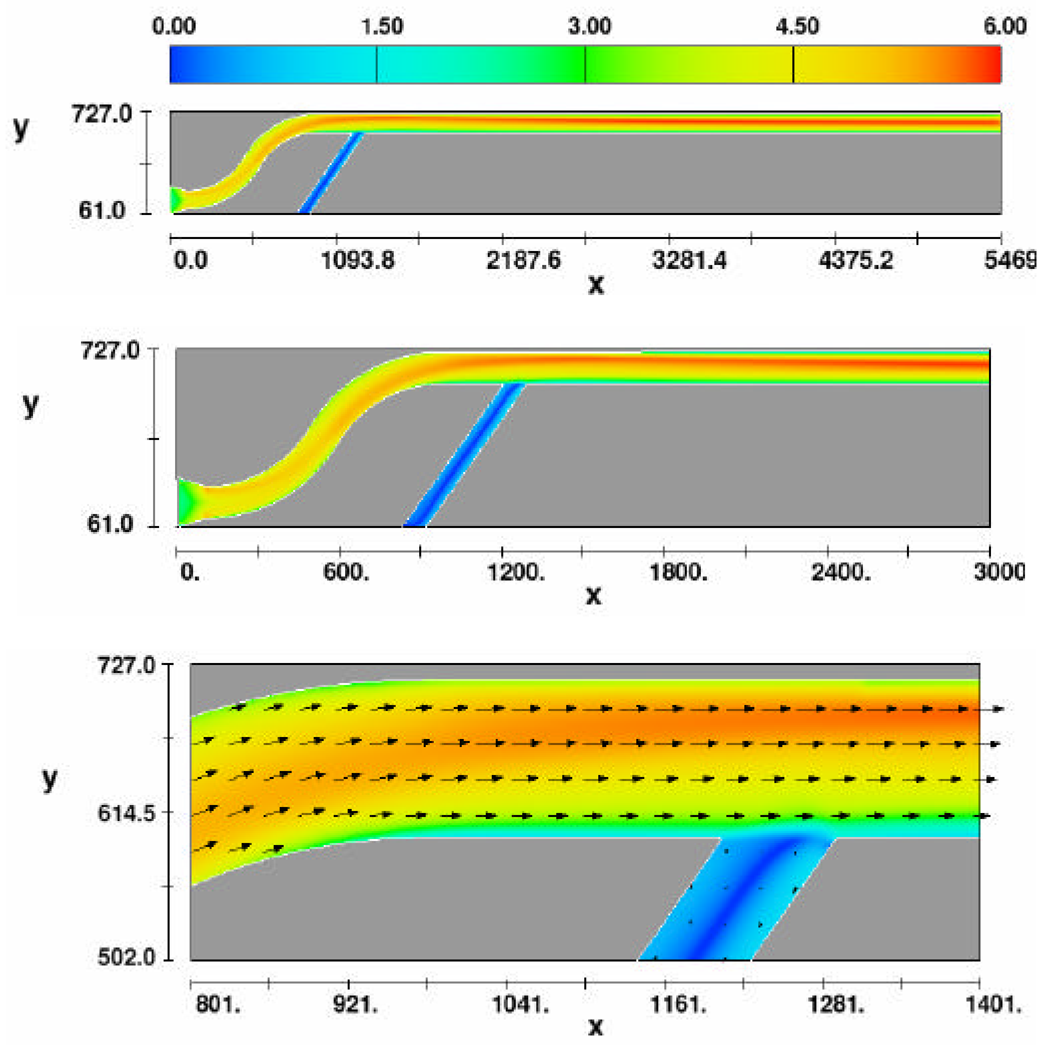


Figure 3. Flow patterns for Case 1. High velocities concentrate near the left bank and angles towards center of the canal. Secondary flow sets up in the Outlet Canal with a stagnant zone near the center of the Outlet Canal. Total velocity color contours are in ft/s at elevation 120.5 feet.

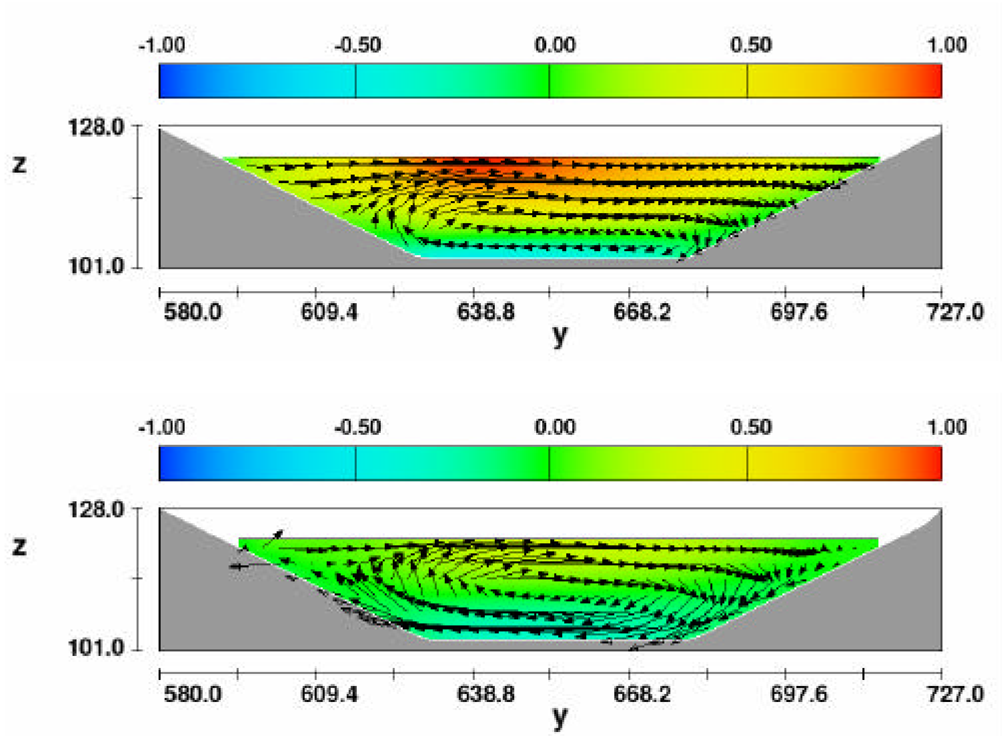


Figure 4. Y-velocity color contours in ft/s upstream and downstream of the confluence for Case 1 at X=945 feet and X= 1,400 feet.

Minor run up on the left bank from the S-shaped bend can be seen in Figure 6. The confluence appears to have little energy loss for this case or effect on major flow patterns.

Due to very little change in the water surface elevation slopes around the confluence, Case 1 results would be nearly identical as to modeling the section without the Outlet Canal.

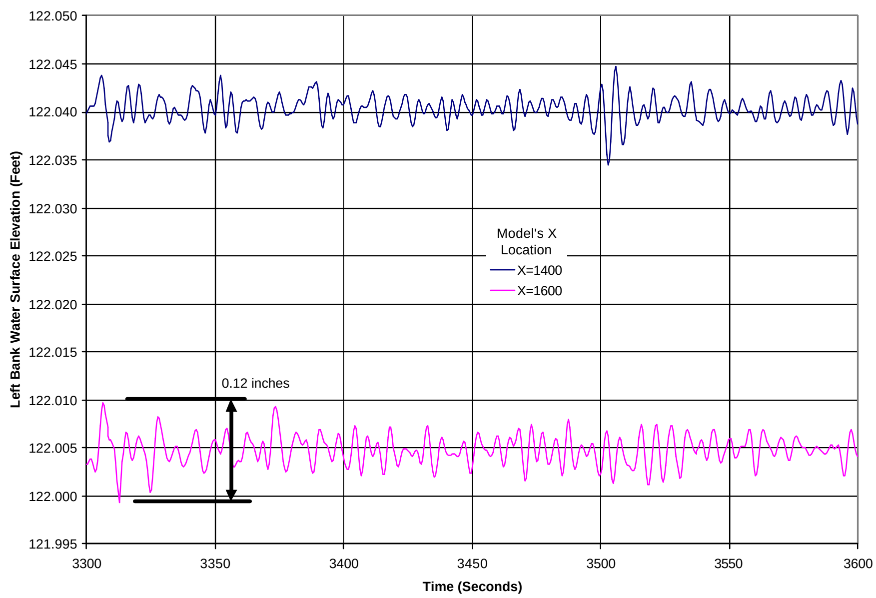


Figure 5. Real time water surface elevations along the left bank for Case 1. The largest peak-to-valley wave was 0.12 inches. This does not include effects of wind.

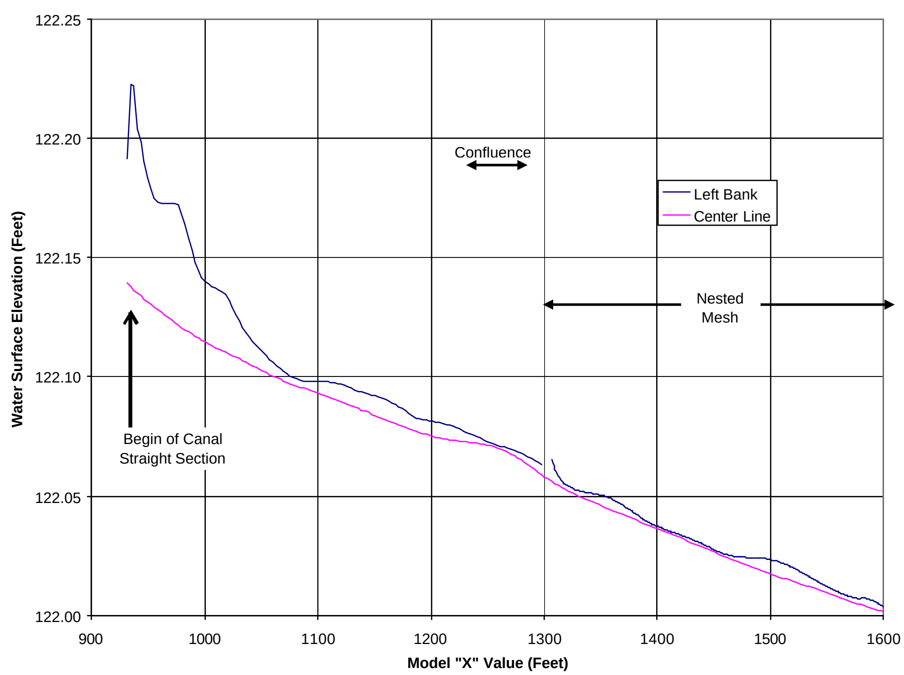


Figure 6. Water Surface Elevations along the left bank for Case 1. The confluence does not create a significant headloss or waves. A simulation without the outlet canal was not needed due to the very little change of water surface slope displayed in this graph. The top of canal lining at the beginning of the confluence (X=1224 feet) is 124.19 feet (at the time of this study).

Case 2 - 5,600 ft^3/s All American Canal with 1,800 ft^3/s Outlet Canal

Case 2 had 5,600 ft^3/s discharging into All American Canal at Drop 2 and 1,800 ft^3/s discharging into the Outlet Canal as shown in Figure 7 and Figure 8.

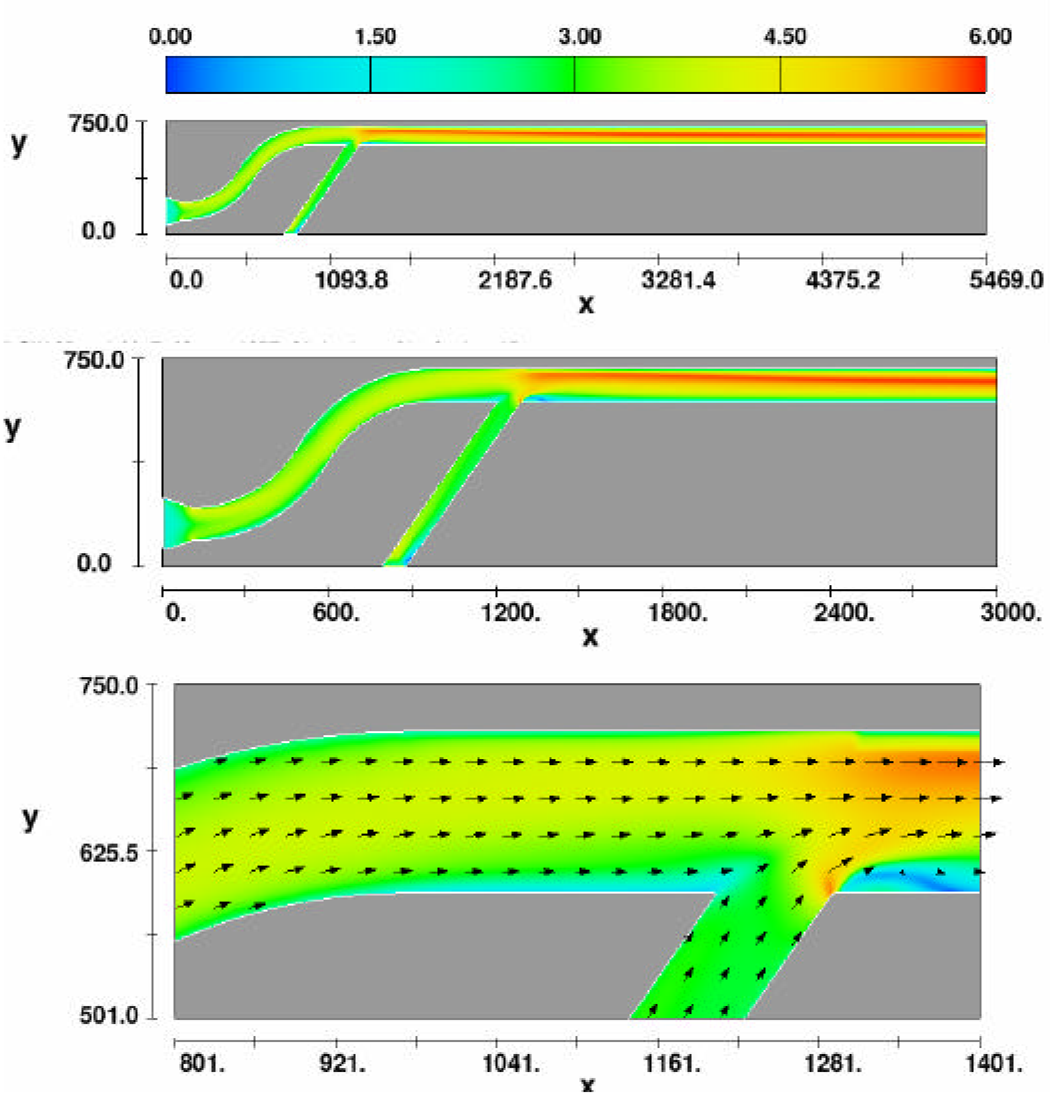


Figure 7. Flow patterns for Case 2. High velocities concentrate near the left bank and angles towards center of the canal. The bottom image shows a flow separation downstream of the confluance with an maximum upstream velocity around 2.5 ft/s. Velocity contours are in ft/s at elevation 120.5 feet.

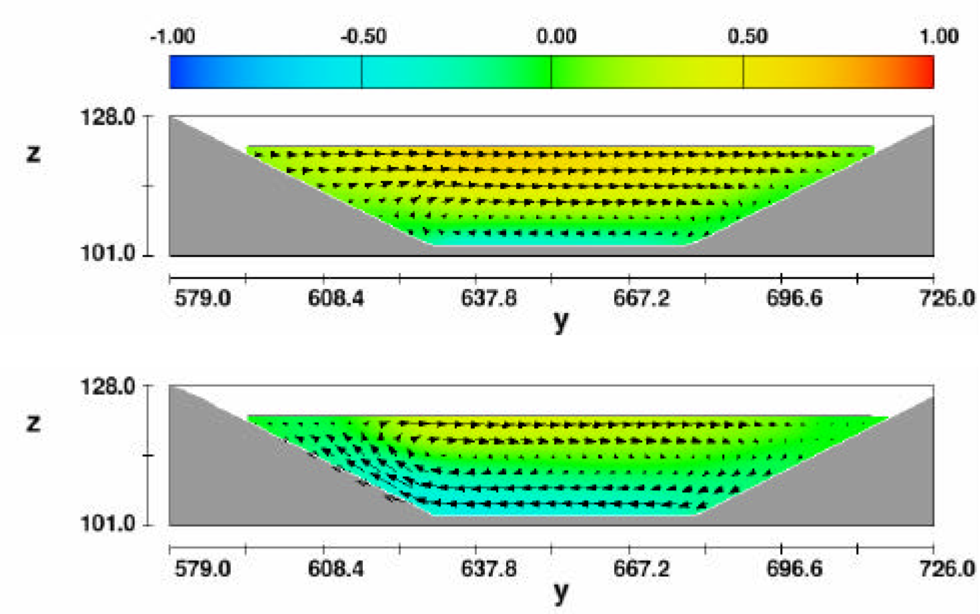


Figure 8. Y-velocity color contours in ft/s upstream and downstream of the confluence for Case 2 at X=945 feet and X= 1,400 feet.

Water surface elevations were recorded at 0.5 second intervals along the left bank at X=1400 feet and X=1600 feet (where the converging flows are expected to have the highest turbulence) and results are displayed in Figure 9. The CFD time step varied around 0.05 seconds. It is noted that the simulation did not include wind or wind effect of wave action.

Case 2 has does not appear to have significant run up on the left bank due to the inflow of the Outlet Canal (Figure 9). However, energy losses and acceleration of flow from the Outlet Canal causes slightly higher water level further upstream (X=900 – end of S-shaped bend) when compared to Case 1 (Figure 6.), the difference of which is around 0.1 feet.

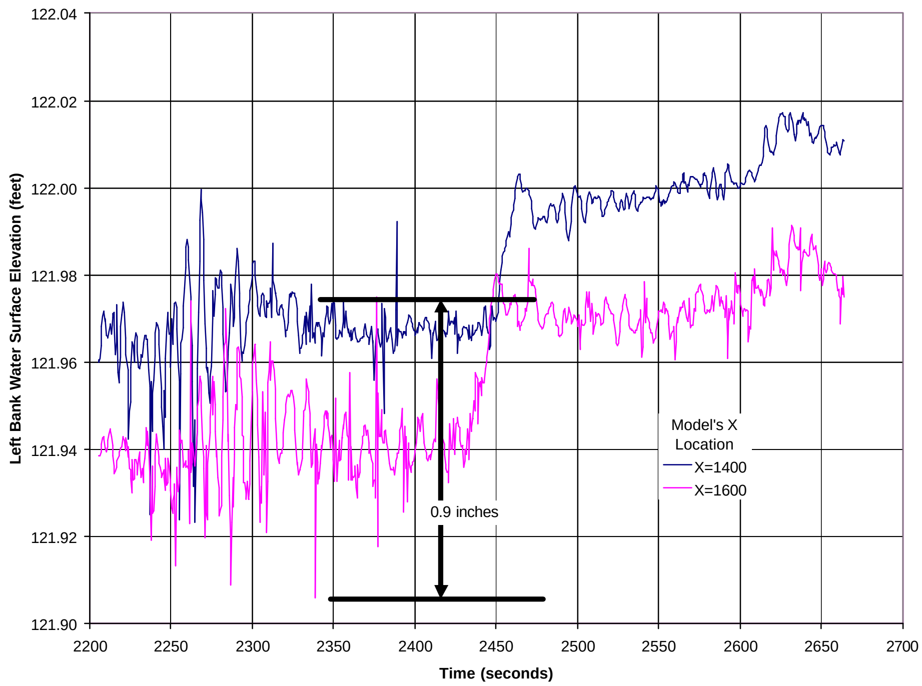


Figure 9. Real time water surface elevations along the left bank for Case 2. A minor low frequency canal transient is still occurring during the simulation around time 2450 seconds and is in the range of 0.72 inches. The largest peak-to-valley wave (excluding the surge) was 0.92 inches. This does not include effects of wind.

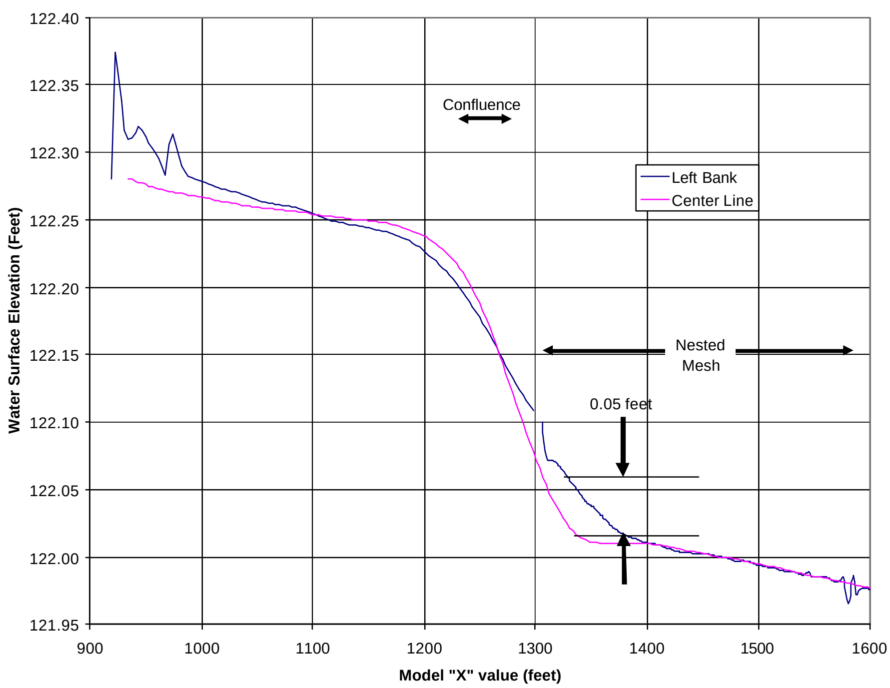


Figure 10. Water surface elevation for Case 2. The top of canal lining at the beginning of the confluence (X=1224 feet) is 124.19 feet (at the time of this study). Run up appears to be 0.05 feet.

Case 3 - 5,600 ft^3/s All American Canal

Case 3 had 5,600 ft^3/s entering the All American Canal and no flow entering the Outlet Canal as shown in Figure 11 and Figure 12. Flow patterns are similar to Case 1 (Figure 3).

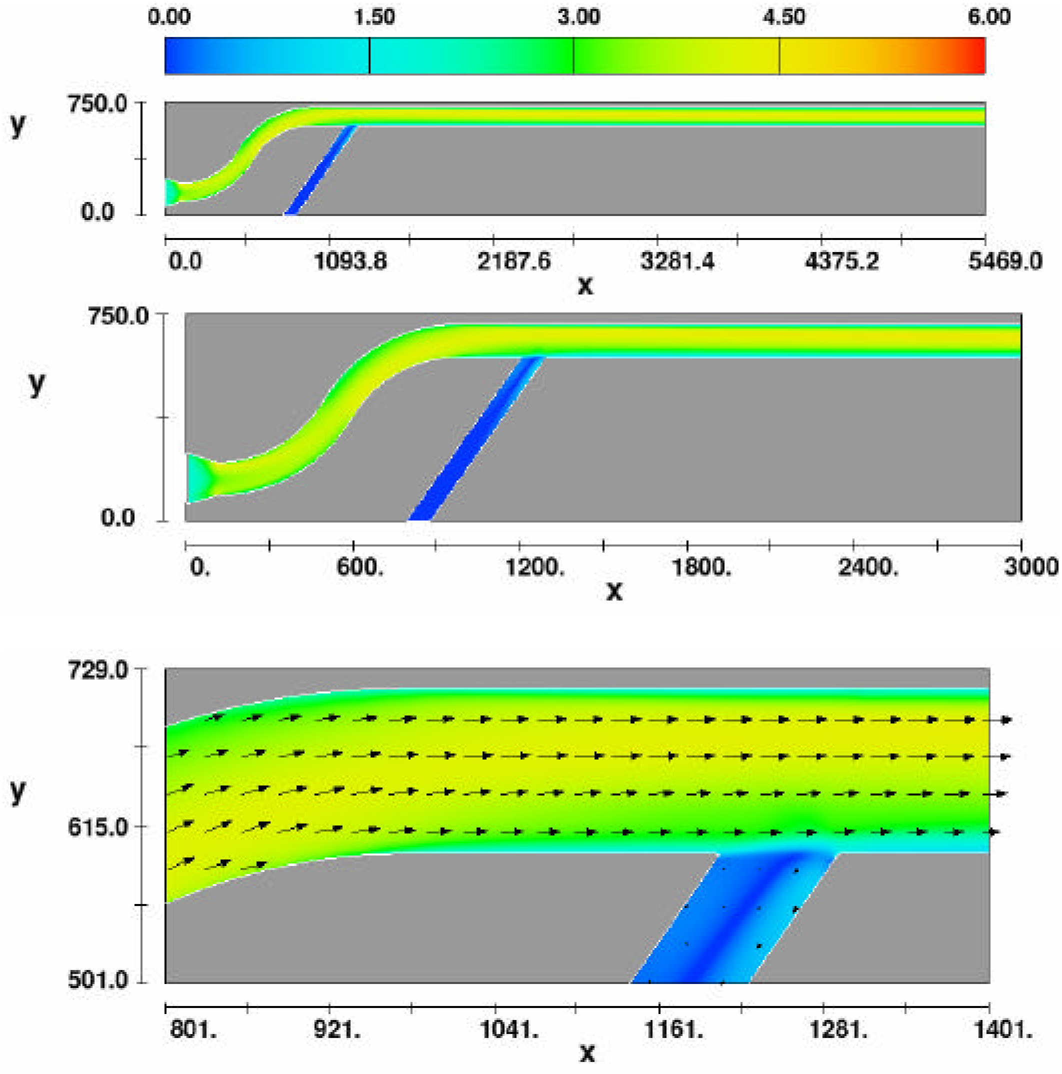


Figure 11. Flow patterns for case 3. High velocities concentrate near the left bank and angles towards center of the canal. Secondary flow sets up in the Outlet Canal with a stagnant zone near the center of the Outlet Canal. Total velocity color contours are in ft/s at elevation 120.5 feet.

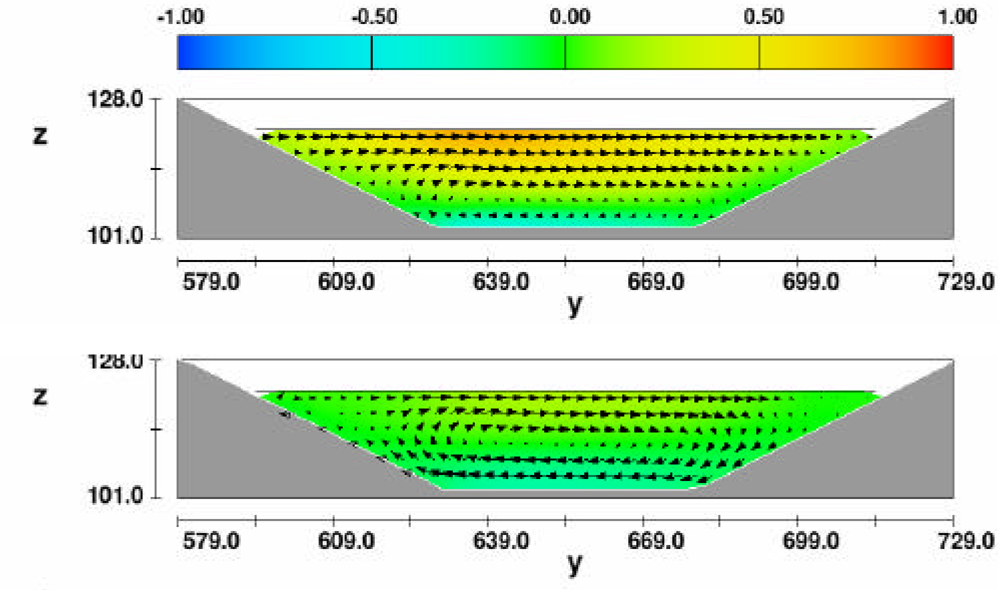


Figure 12. Y-velocity color contours in ft/s upstream and downstream of the confluence for Case 3 at X=945 feet and X= 1,400 feet.

Water surface elevations were recorded at 0.5 second intervals along the left bank at X=1400 feet and X=1600 feet (where the converging flows are expected to have the highest turbulence) and results are displayed in Figure 13. The CFD time step varied around 0.07 seconds. It is noted that the simulation did not include wind or wind effect of wave action.

For case 3, minor run up on the left bank from the S-shaped bend can be seen in Figure 14. The confluence appears to have little energy loss for this case or effect on major flow patterns.

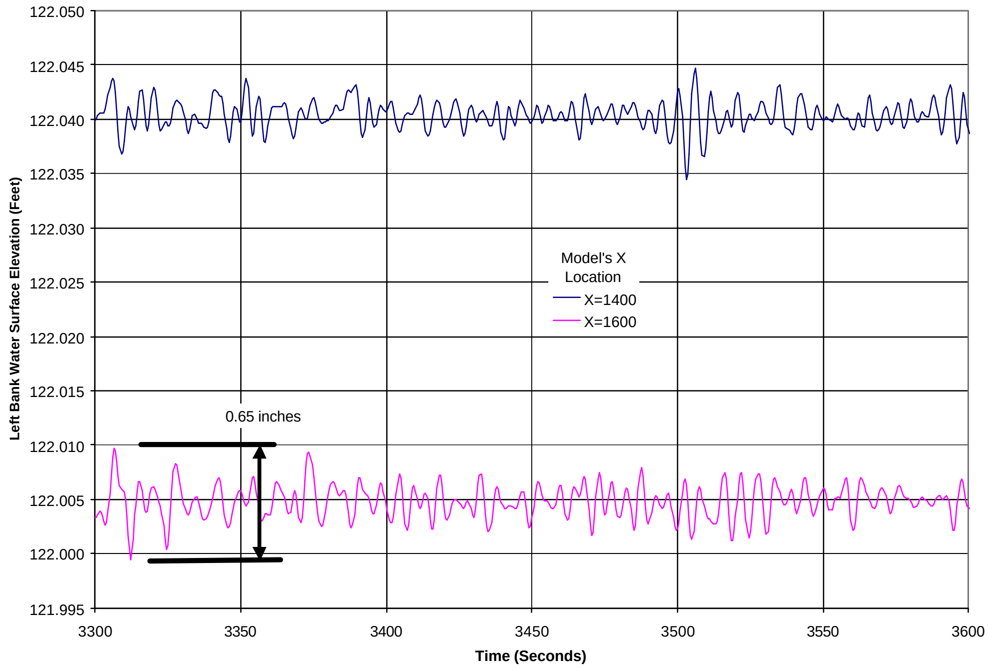


Figure 13. Real time water Surface Elevations along the left bank for Case 3. The largest peak-to-valley wave was 0.6 inches. This does not include effects of wind.

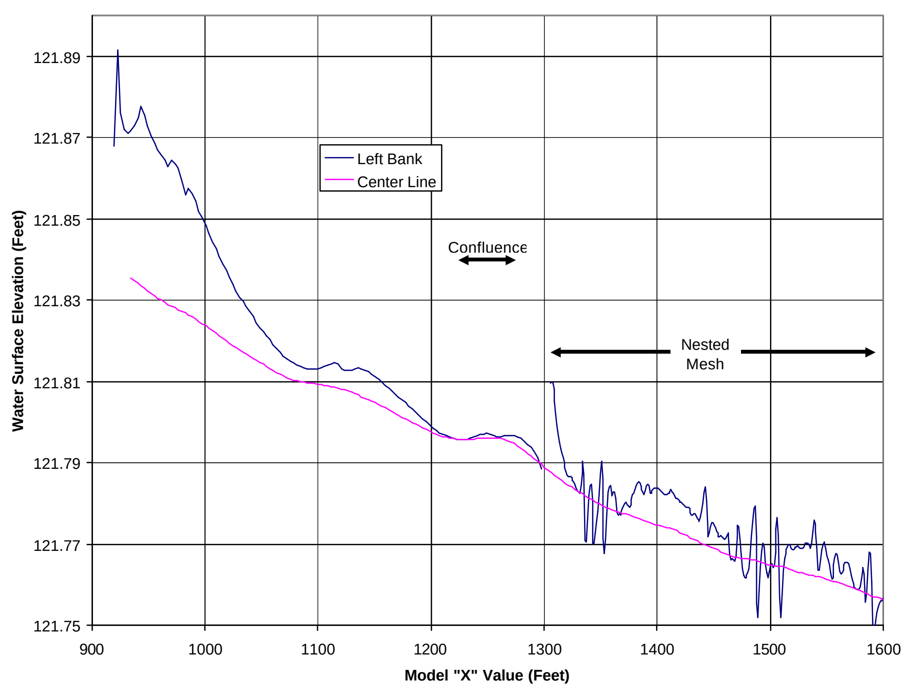


Figure 14. Water Surface Elevations for Case 3.

Case 4 - 1,800 ft^3/s Outlet Canal

Case 4 had no flow entering the All American Canal and 1,800 ft^3/s entering the Outlet Canal as shown in Figure 15 and Figure 16.

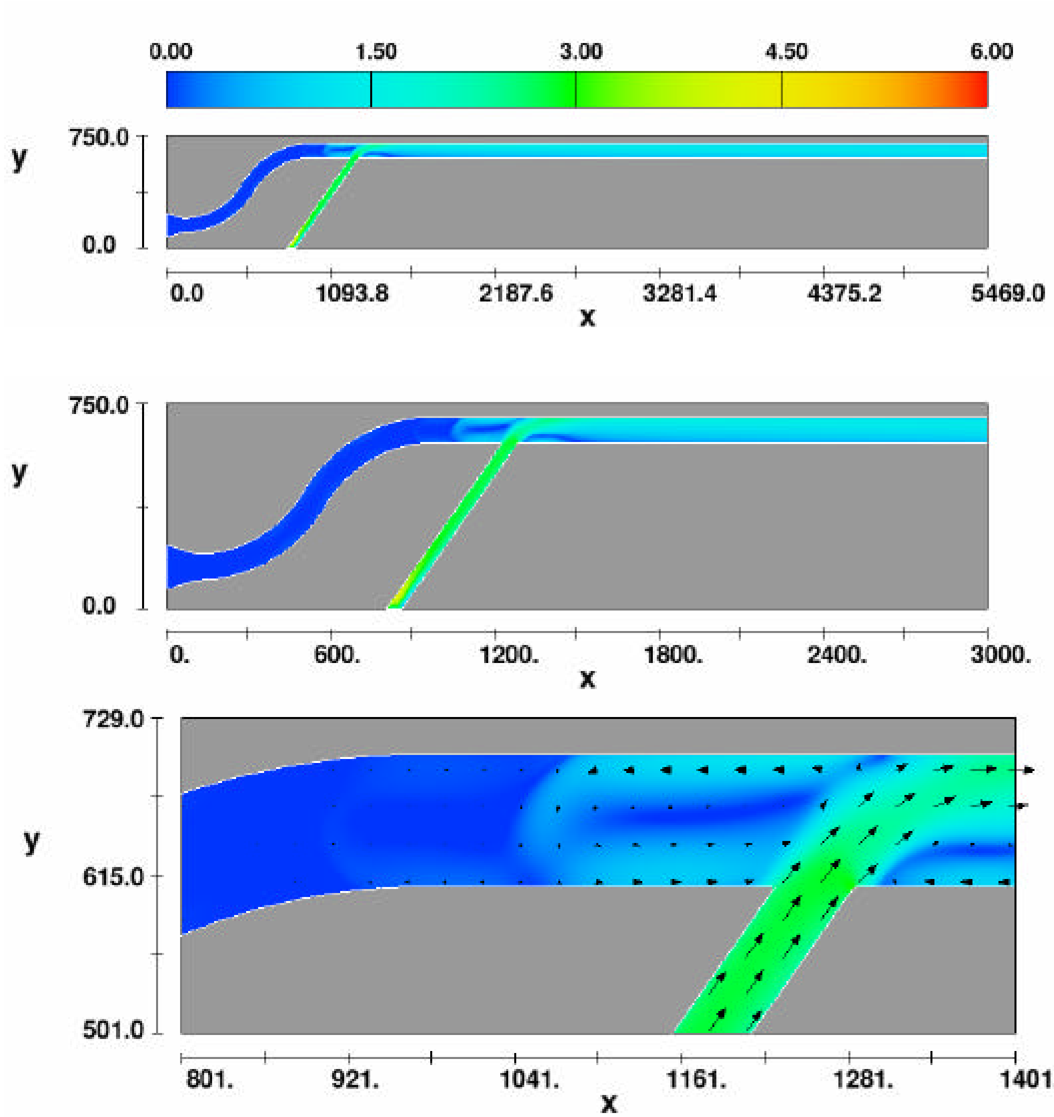


Figure 15. Flow patterns for Case 4. High velocities concentrate near the left bank and quickly spreads across the canal. The bottom image shows a flow separation downstream of the confluence with an maximum upstream velocity around 1.0 ft/s. Velocity contours are in ft/s at elevation 120.5 feet.

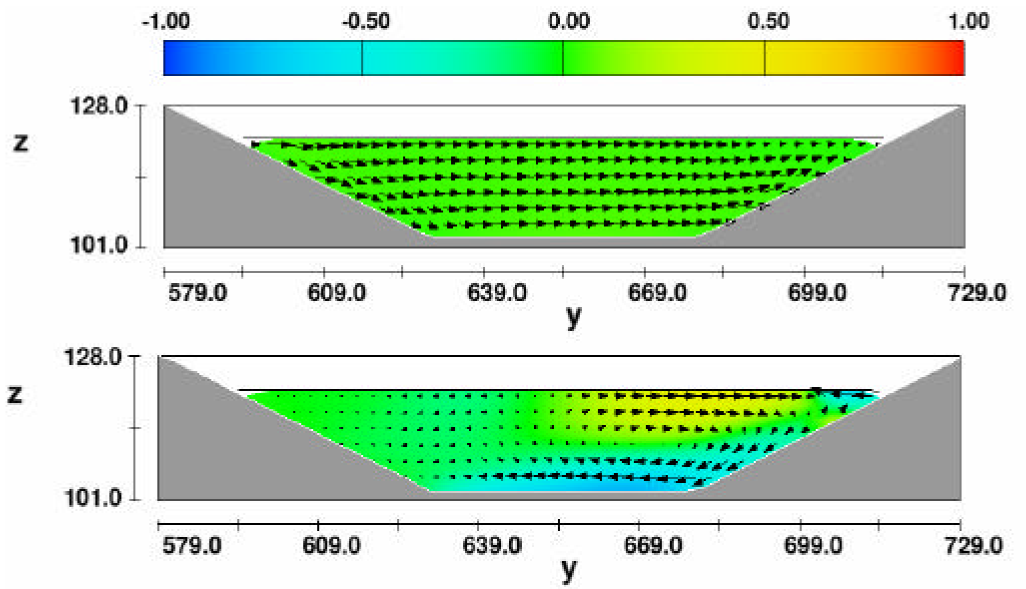


Figure 16. Y-velocity color contours in ft/s upstream and downstream of the confluence for Case 4 at X=945 feet and X= 1,400 feet.

Water surface elevations were recorded at 0.5 second intervals along the left bank at X=1400 feet and X=1600 feet (where the converging flows are expected to have the highest turbulence) and results are displayed in Figure 17. The CFD time step varied around 0.06 seconds. It is noted that the simulation did not include

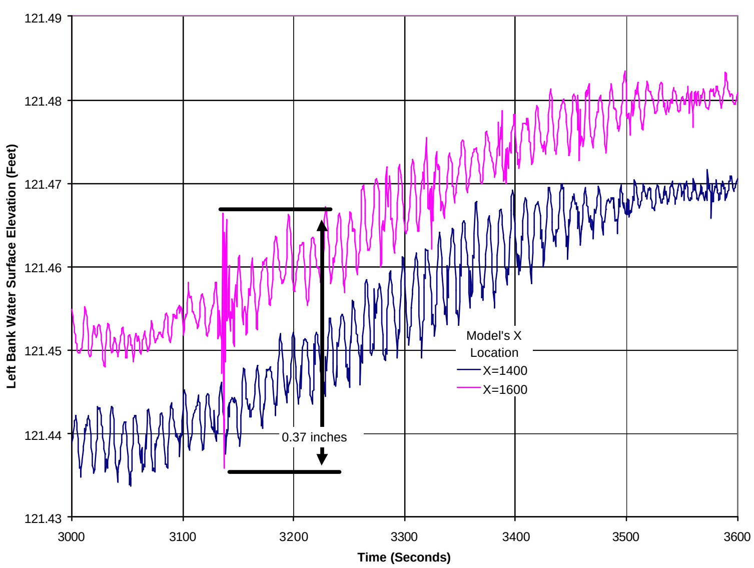


Figure 17. Real time water surface elevations along the left bank for Case 4. A minor low frequency canal transient is still occurring during the simulation and is in the range of 0.48 inches. The largest peak-to-valley wave (excluding the surge) was 0.36 inches. This does not include effects of wind.

wind or wind effect of wave action.

Case 4 does not appear to have significant run up on the left bank due to the inflow of the Outlet Canal (Figure 18). Peak left bank water surface elevation is 121.48 feet where as the peak elevation in that section for case 2 was greater than 122.2 feet.

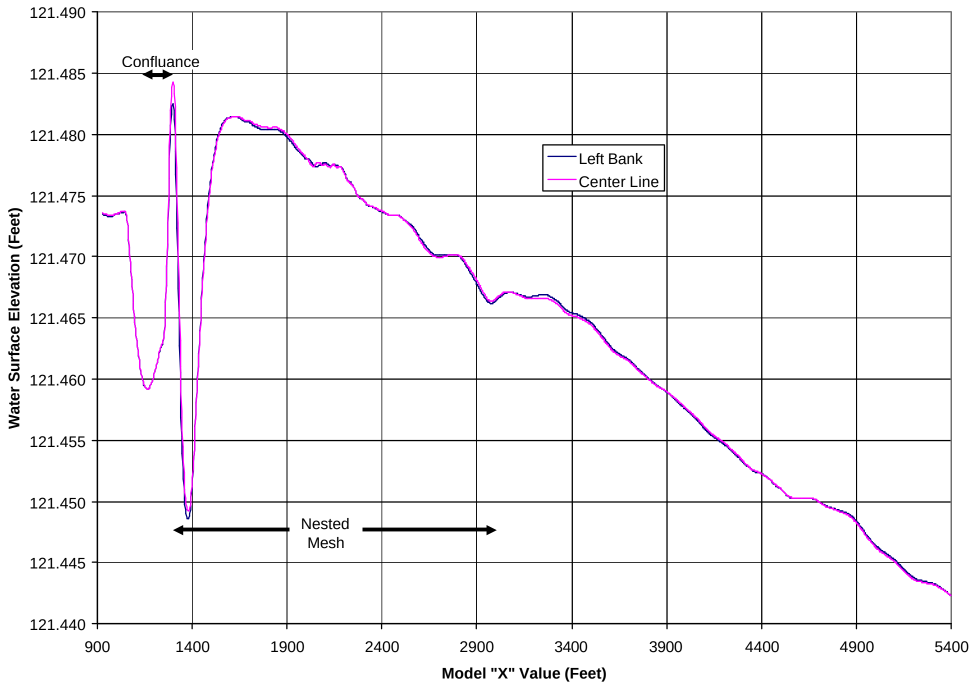


Figure 18. Water Surface Elevations for Case 4.

Headloss

Water surface elevations are reported in Table 3 for all simulated conditions. These results can be compared to HEC-RAS simulations and standard design methods to determine if the top of canal lining is appropriate. It may also be used to check power generation assumptions. Additional headloss due to gate operations or power generation are not included in the table.

Energy loss due to the confluence without Outlet Canal flow appears insignificant. The additional head requirement for the Outlet Canal flowing at maximum and with the maximum downstream canal discharge appears to be 0.1 feet, which is the difference of Case 1 and 2.

Table 3. Water surface and change of water surface elevations for the simulated cases. Change of water surface elevation attributed by confluence operations was calculated by comparing the Drop 2 outfalls values for case 1 and 2. Simulations did not include Drop 2 losses.

	Water Surface Elevations			Change of Water Surface Elevations	
Case	Drop 2 outfall (feet)	Siphon outfall (feet)	End of Model (X=5,469 feet) (feet)	Drop 2 outfall to end of model (feet)	Siphon outfall to end of model (feet)
1	122.4	122.1	121.4	1.0	0.7
2	122.5	122.3	121.4	1.1	0.9
3	122.1	121.8	121.4	0.7	0.4
4	121.5	121.6	121.4	0.1	0.2

Safety Considerations

For cases 1, 2, and 3 the rotational flow conditions displayed in Figure 4, Figure 8, and Figure 12 is predominately downstream of the S-shaped bend. This is characteristic of standard secondary currents generated by open channel flow through a bend. Upwelling can be seen along the bottom-right of the canal invert with a minimum 0.2 ft/s that may help lift swimmers. A very small amount of down welling can be seen along the left bank that would have a weak submergence effect on swimmers.

Case 4 shows the greatest submergence effect along the left bank with downward flow around 0.8 ft/s.

A ladder or ladders placed along the sides would be beneficial.

Sedimentation

Case 1

Sedimentation is probable in the Outlet Canal while 7,400 ft^3/s flows in the All American Canal. The highest amount of sedimentation may happen where the dark blue color contour (0 ft/s) can be seen in Figure 19; however the amount and nature of the sedimentation cannot be predicted by this study. This sediment is likely to scour out during Case 2 and 4 operations. The minimum velocity in the All American Canal observed for this condition was 3.5 ft/s, which may scour any sediment deposited during other operations.

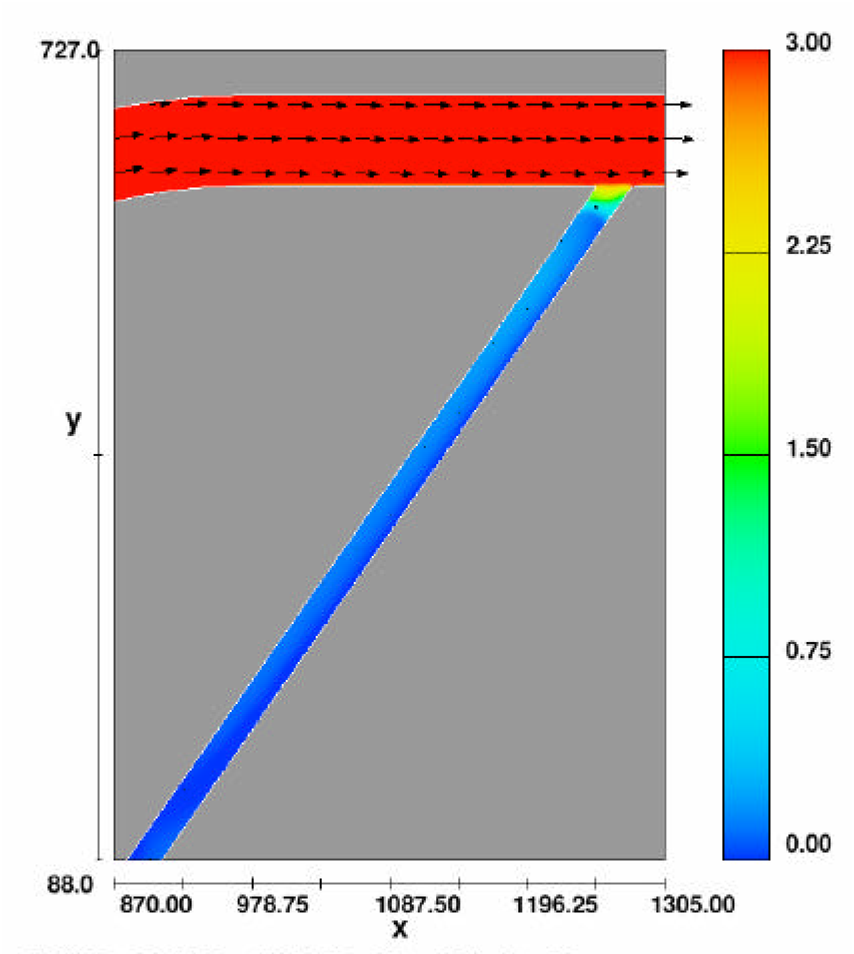


Figure 19. Velocity contours in ft/s at elevation 108.5 feet for Case 1. The highest amount of sedimentation may happen in the area of the dark blue color contour (0 ft/s). The minimum velocity in the Main Canal observed for this condition was 3.5 ft/s, which would be available to scour any sediment deposited during other operations. The upper bound of the velocity contours displayed was limited to 0.5 ft/s.

Case 2

Significant sedimentation is not probable for Case 2. The minimum velocity was observed in the Outlet Canal and was 1.4 ft/s, while the minimum in the All American Canal was above 2.5 ft/s. The separation zone that can be seen in Figure 7 does not extend to the canal floor as can be seen in Figure 20 and should not facilitate sediment build up.

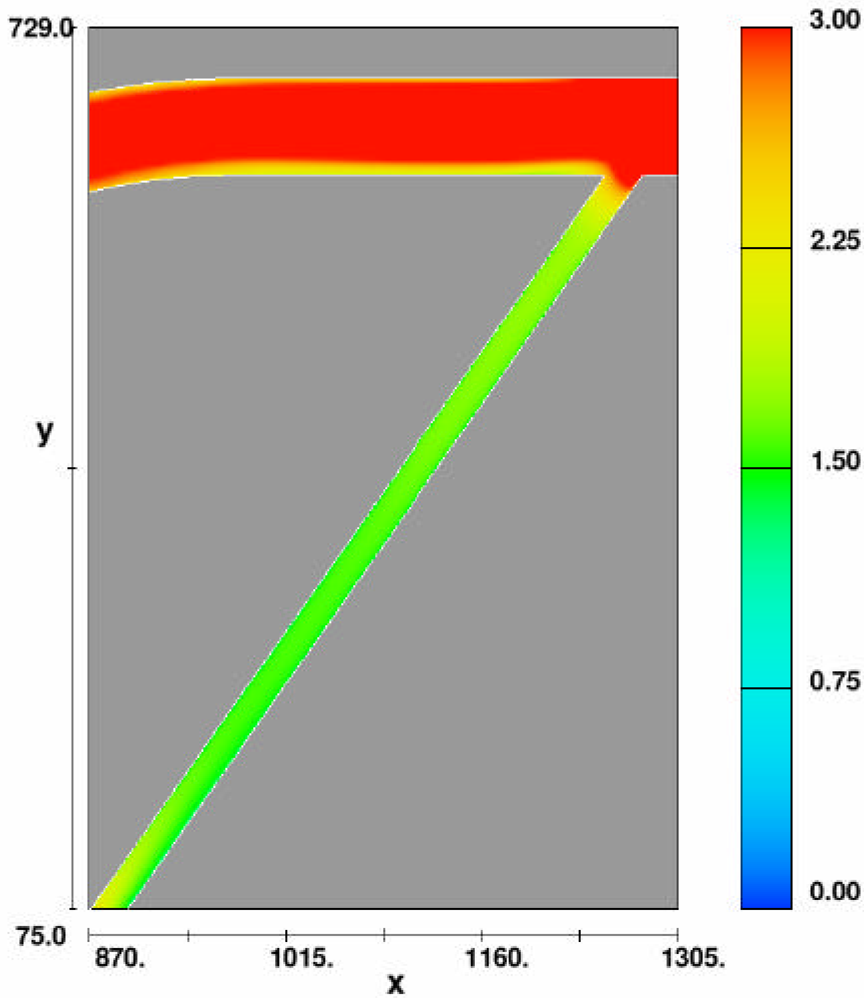


Figure 20. Velocity contours in ft/s at elevation 108.5 feet for Case 2. The minimum velocity on the bottom of Outlet Canal is above 1.4 ft/s while the All American Canal is above 2.5 ft/s. The upper bound of the velocity contours displayed was limited to 3.0 ft/s.

Case 3

Sedimentation is probable in the Outlet Canal while 5,600 ft^3/s in the All American Canal. The highest amount of sedimentation may happen where the dark blue color contour can be seen in Figure 21, however the amount and nature of the sedimentation cannot be predicted by this study. The minimum velocity in the All American Canal observed for this condition was above 3.0 ft/s.

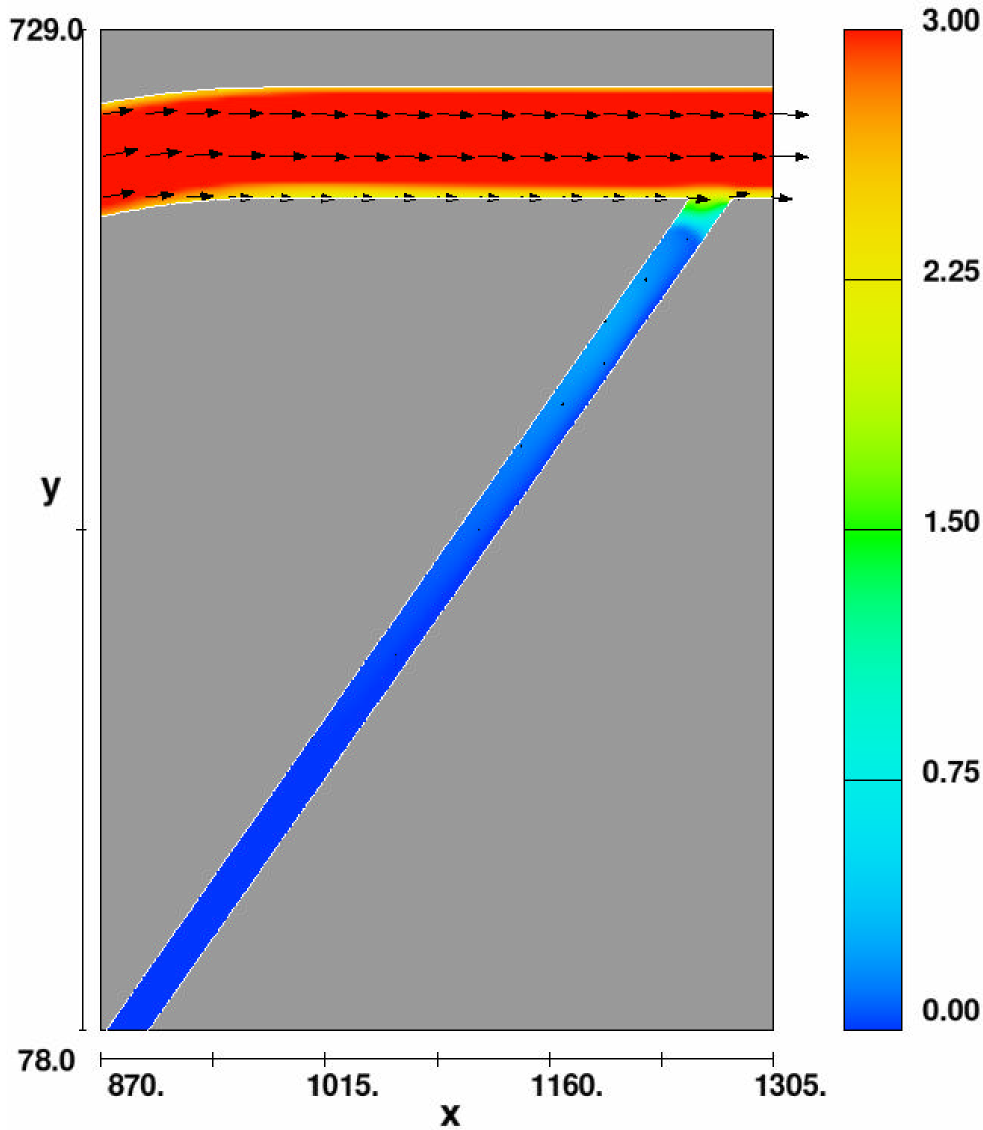


Figure 21. Velocity contours in ft/s at elevation 108.5 feet for Case 3. The highest amount of sedimentation may happen in the area of the dark blue color contour (0 ft/s). The minimum velocity in the All American Canal observed for this condition was 3.1 ft/s, which would be available to scour any sediment deposited during other operations. The upper bound of the velocity contours displayed was limited to 0.5 ft/s.

Case 4

Since the storage reservoir is designed to drain in 3 days or less, significant sedimentation of the All American Canal is not a concern. Figure 22 shows flow patterns in the confluence from this condition.

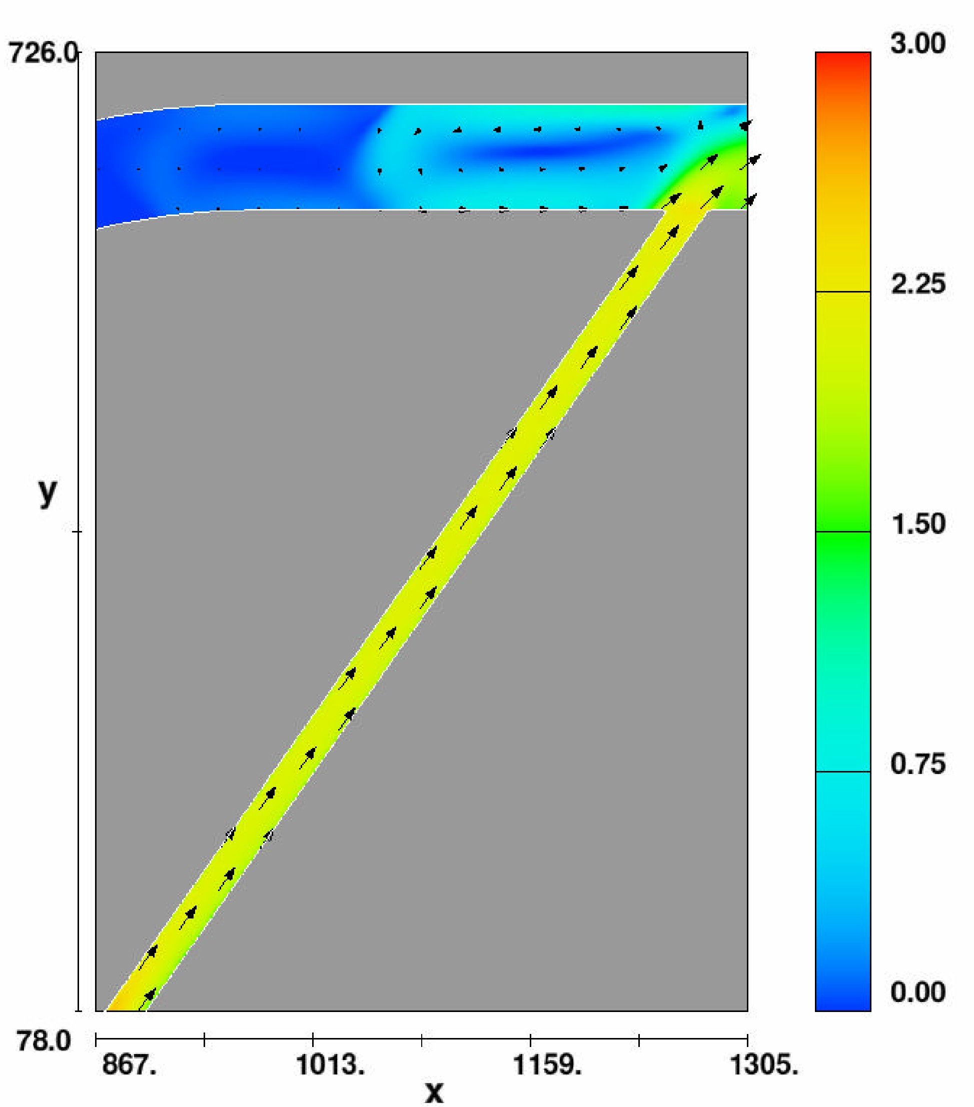


Figure 22. Velocity contours in ft/s at elevation 108.5 feet for Case 4. The minimum velocity on the bottom of Outlet Canal is above 1.4 ft/s while the downstream portion of the All American Canal is nearly 1.0 ft/s. The upper bound of the velocity contours displayed was limited to 2.1 ft/s.

References

i Drop 2 Storage Reservoir Feasibility Design Report, Colorado River Front Work and Levee System, California, Technical Service Center, Reclamation, 2006.
ii Flow Science Inc., FLOW-3D version 9.0 User's Manual. 2005.
iii C.L.M.H. Navier, M´emoire sur les lois du mouvement des fluides, M´em. Acad. Sci. Inst. France 6 (1822), 380–440.
iv G.G. Stokes, On the theories of internal friction of fluids in motion, Trans. Cambridge Philos. Soc. 8 (1845).
v J.M. Sicilian, "A FAVOR Based Moving Obstacle Treatment for FLOW-3D," Flow Science, Inc. Technical Note #24, April 1990 (FSI-90-TN24).
vi Michael R. Barkhudarov, "Semi-Lagrangian VOF Advection Method for FLOW-3D," Flow Science, Inc. Technical Note 63, July 2003 (FSI-03-TN63)